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IMPLEMENTATION OF AUTOMATION PROCESS IN GENERATING CAD MODELFOR RIM WHEEL

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ABSTRACT

The purpose of this study is to develop a Computer Aided Drawing (CAD) program that allow automation process in CAD software to generate three-dimensional models automatically. Automation process has been widely used and received give positive feedback in various industries due to its convenience in drawing process and time saving. It also provides high reliability, more accurate and also allow multiple task in design creation. The process consists of two stages which are the construction process of rim wheel model using Computer Aided Three-dimensional Interactive Application (CATIA) software and the model automation coding development process. The rim wheel model was constructed and recorded in CATIA Macros tool. The recorded data was then utilized to develop the coding script for the model automation process using Visual Basic language. The important parameters were identified and edited in the coding script to generate the basic shape of the model and other features. The final model supposed to have an ability to alter into three different rim wheel design with different parameters and dimensions. This program could be used as the future platform to fulfill specific customer requirement of a desired product.

Keywords: automation, CAD models, CATIA, visual basic, rim wheel.

INTRODUCTION

Computer aided design (CAD) software have become a mainstream media in most industry as a tools for project designing for the past 20 years. The software has rapidly evolving and continuously improve to become a programs with enormous capabilities and function(Lowe & Hartman, 2000). CAD software always been related with the automation since the beginning history of CAD as creating and editing a Two-Dimensional (2D) and 3D model is faster when using a computer compared to manual hand drawing. Most user still desperately manually doing the same bland CAD translation and repair work without consider how despite intuitive and advance the development of the existing CAD system. Because of that, the application of CAD automation can help to ease the operating process by the user (Venkatesan & Karnan, 2009).

The concept of CAD automation enables the user to automatically created the design or adjusting the method and process of the already made design to fulfil the criteria of the new design. The process allows the user to automate translation and repair processes of the design that are already in large part by enabling users to translate multiple files at one time without having to open or view them graphically (Ramly *et al*, 2013).

There are two method that commonly used for CAD automation which are the manually writing a complete programming code and the application of macro tool in the modeling software. Depending on the applications and the requirement for the desired model, a particular method that is suitable is used (Weisenberger, 2012).

Instead of repeatedly creating the similar models and nearly 80% of design activities have a routine nature, engineers should be able to create automation program so that it automatically creates the new CAD models or to generate the further modification of the same model (Zbiciak *et al*. 2015). The automation process is meant to help in reducing the development time of a product especially in mass production industry. The process not only saves time, but also increases quality of data input for a model and minimizing the possibility of human errors. Thus, this criterion will lead to increasing in terms of productivity (Siddesh & Suresh 2015).

Previous study by Wayzode & Tupkar (2012) used CATIA V5 macro programming for shaft coupling. It can be used to draft different views of the coupling which can increase productivity of the designer, increase the quality of design and reduces lead time for design of shaft coupling. Ramly *et al*. 2013 study an automated a complex computer aided design concept generated using macros programming for commercial aircraft. It found that the advantages of macros programming allowing flexibility for design exploration and increase the usability of the design solution.

Nowadays, online purchasing has become the current trend for the consumers. Almost 75% of the customers (from 756 customers in USA, Germany and China) expressing a willingness to purchase a vehicle online (Capgemini consulting, 2016). However, specific demand from consumers may cause problems and delays in the production process as different design are required to fulfil their needs. Therefore, the automation are needed to facilitate the ordering process of a product based on specific customer requirement.

For an example, a shop that using the automation process can create a simple form from the application of CAD automation where the customer can order their product easily through online or personally at the shop. The customer are only needed to put in the parameter that they wanted in the order form and the design of the product will be automatically created in the CAD and the

can see the model or in assembly mode. This can be apply in various type of products that need specific requirement and features. For example such as the grill in the front car's bumper, car bumper, spoiler, rim wheel, car lamp and etc. This paper presents the procedure to develop automation process to generate the CAD models for rim wheel (as a guideline/platform) using the macros programming software in CATIA.

METHODS

A. Automation methodology

Three different designs of car rim wheel were modelled for the automation process based on the real rim design (Chang & Yang 2009, Ishikawa S *et al*. 2014). Important parameters and features of the wheel were identified and used to construct the 3D model of the wheel and applied in automation process. Figure-1 shows the flow chart of the automation process in CAD.

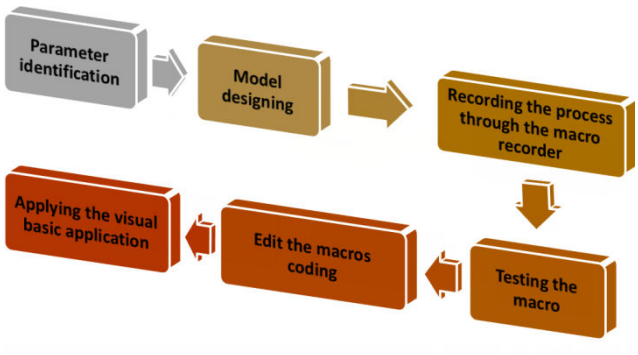


Figure-1. Flow of the main process of automation in CAD.

B. Macro programming in CATIA

A macro is a text string that combines commands and options for repeated use in CATIA. All inputs, outputs and necessary supporting data were required and used to develop a macro programme. The programing script for the automation process was written in Visual Basic for Applications (VBA). One of the important tool in macro programming is the macro recorder where the geometrical model of the wheel was easily adjusted by referring to the initial model that has been recorded. Parameter interface was added so that some of the dimensions and features can be changed directly.

a) Design parametric identification

Actual rim wheel was used to construct the 3D geometrical model which will be used as a main reference. Four common parameters and features were considered to be varied in the automation process which was the rim diameter; rim width, number of bolt hole and number of rim spoke as shown in Figure-2. Details of the parameters and features of three rim wheel designs are tabulated in Table-1.

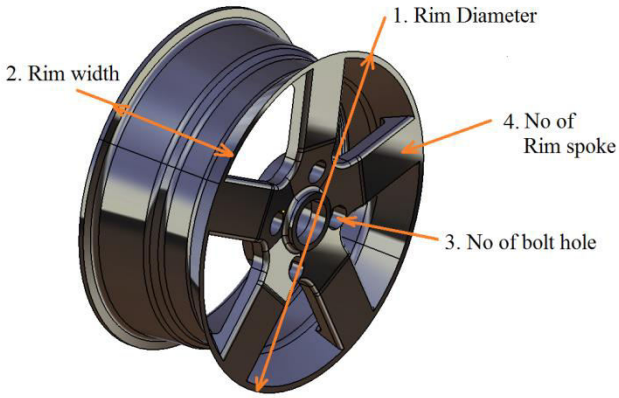


Figure-2. Parameters and features of rim wheel model for automation process.

b) Macro programming procedure

The macro programming was edited by adding a dialog box functions into the programing script. These dialogue box are categories into input-box for input data and message-box for output data. These dialogue boxes were used to create a pop-up message box to insert the desired parameters and features of the rim wheel.

The command task can be compiled into one form to insert the parameter value using *Userform* in macros coding. It can be access through the visual basic editor in command box: *MyForm.Show*. The function of the macro programme was verified to check the automation models were constructed with the input parameters.

RESULTS AND DISCUSSIONS

A. Development of coding for 3D model of the rim wheel

In order to make the modelling process of a model sample able to be automated in CATIA, a set of coding logarithm for the model need to be defined in the first place. To get the coding script, the modelling process need to be recorded by using the function of start recording in the Macros tools. The macro will be the module for the coding reference object.

The modelling process must be fluent without using the undo or delete button while recording in macro to avoid the complexion of algorithm in the coding. Once the drawing finish, stop the recording before exiting the drawing thus all the data from the start of the drawing until finish will be stored in the Macros. Three design of different car rims has been choosing to be created as the model as shown in the Figure-3.

The coding script of these models are stored in the Macros library and can be run again to automatically create the same model by selecting the code in the library and click the 'run' button. These programming script will be edited to be use for the later in depth process of automation. This coding script function is to declare the acquired code through the recording Macros function. The recorded code is important in order to create a relation

with the coding from the *Userform* later on so that the automation process of the model can be done successfully. Also, the parameter from this code will be the object references for the *Userform* coding.

B. Development for the application of *Userform*

The application of the *Userform* in Visual Basic is one of the main roles to execute the process of CAD automation for CATIA V5. The *Userform* can be created from the Visual Basic Editor which can be access from the toolbar in CATIA. For each application assigned to the *Userform*, the code need to be put in manually to make it function. The coding developed in the *Userform* is the key item used to link the logarithm between the coding of the *Userform* and the coding of the model. The interface shows an empty form to be fulfilled by the user to complete the automation process. For this project, the *Userform* used are divided into two phase which is the first phase is to choose the design and the second one is for the parameter input of the design.

The *Userform* entitled *Rim Design* will shows the picture of different selection of the available rim design with a command button assigned to each picture. At the bottom of the *Userform* there will be a command button with a 'cancel' function. Figure 3 shows the image of the *Rim Design Userform.*

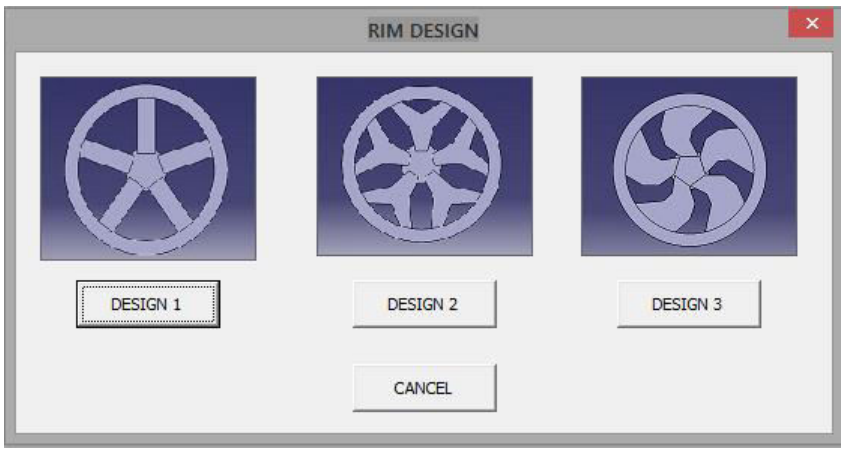


Figure-3. *Userform* for design variety (first phase).

For the second phase, the form will include empty textbox with the description for the parameter requirement as shown in Figure-4. It also includes two command button with the function to cancel and create. To make the data input can be use into the model, the coding line for parameter of the model need to be changed. The value of the parameter in the coding are replaced with the *textbox* command function. Thus the value for the parameter can be read and gain from the value put in the textbox of the *Userform*. For example, taken the coding sample from the design one as shown from the previous topic, the original coding value for the rims diameter is *length1.Value = 431.8 (for example).* From this line, the number can be change to the *textbox1* command which is the empty textbox for the rim diameter value. This will allow the value of the rim diameter to be altered depends on the input value from the textbox.

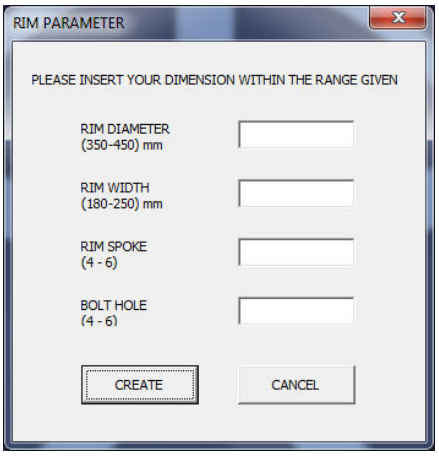


Figure-4. Form for the parameter input (second phase).

c. Simulation and result of the program

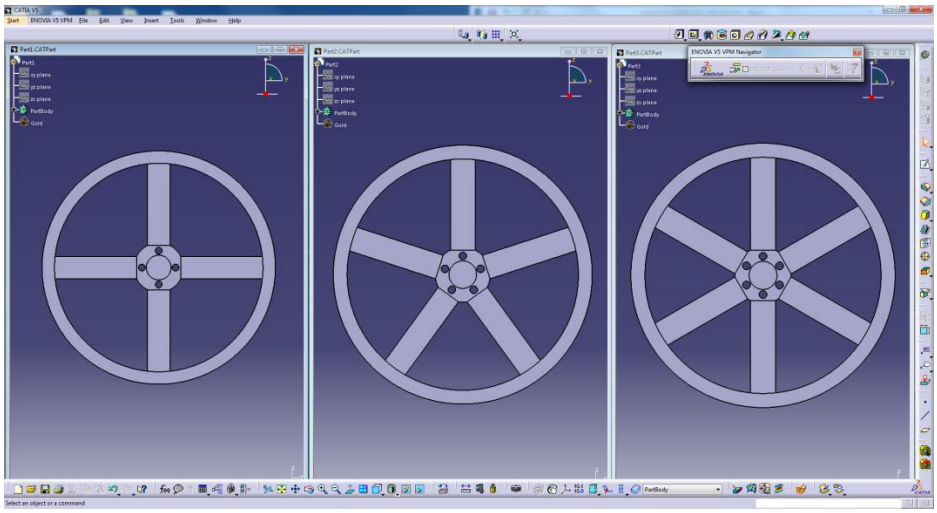
When the Macros is executed, a form entitles *RIM DESIGN* will pop out to show the menu for the selection of the three available design as visualize in Figure-4. Clicking either *DESIGN 1, DESIGN 2* or *DESIGN 3* button in the form, will close the form and result in another form to shows up. This new form will show a list of empty text box with the parameter description that need to be filled in. The value for the input must be the number within the range as given in the form. When the desired parameter of the design has been filled, clicking on the create button will create the design

corresponding with the value that has been filled in the textbox.

To shows the flexibility of the created Macros program, the different parameter value for design 1 is set to be used into the program as shown in the Table-1. The result for the object created from this value is shown in Figure-5. Some of the example of the possible rim design that can be created with this program is shown in the Figure-6. This result proves the logarithm of the automation is function successfully.

Table-1. Different parameter of design 1.

No.	Parameter characteristic	Design 1 (A)	Design 1 (B)	Design 1 (C)
1	Rim diameter	430mm	440mm	450mm
2	Rim width	230mm	240mm	250mm
3	Number of bolt hole	4	5	6
4	Number of rim spoke	4	5	6



(a) (b) (c)

Figure-5 (a), (b) and (c). Example of generating a model with different parameter.

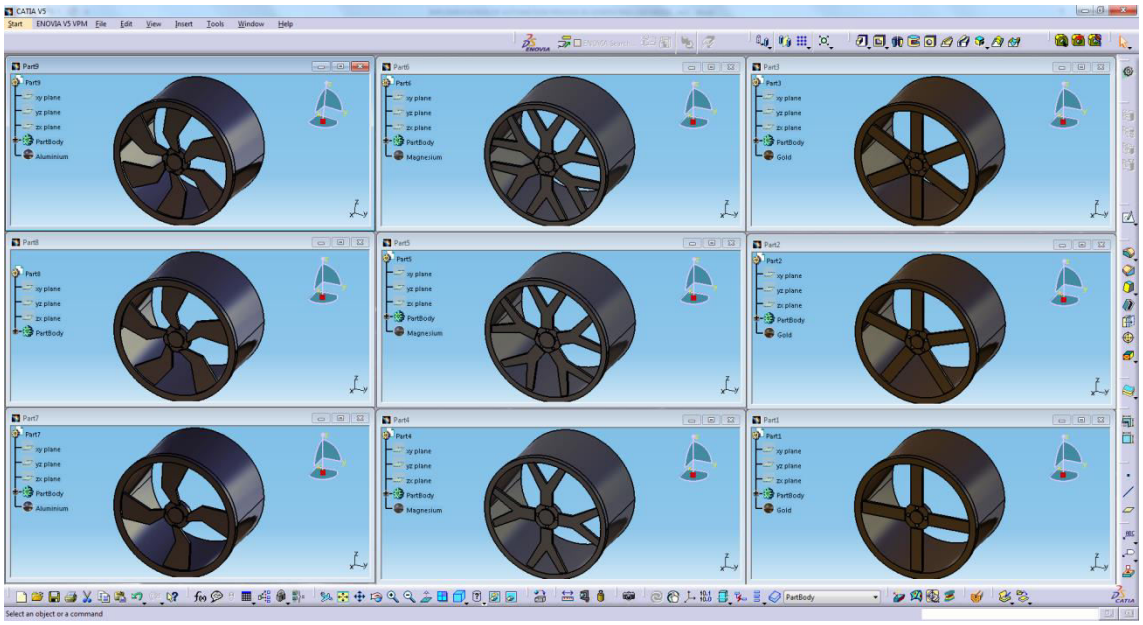


Figure-6. Result for the different design and parameter.

CONCLUSIONS

As a conclusion to this project, Macros application and visual basic programming played a huge part on developing a CAD automation process in CATIA software. The automation processes are meant to help in reducing the development time of a product especially in mass production industry and also minimizing error. Macros tools is one of the system for automation process development that is useful for future industrial references. The Macros recording function allow the repetitive process of drawing a model to be done without the needs to draw the same model over again and again.

The important aspect that needs to be considered in making the automation process possibly done is the method of parameter identification of the model. This is the key item to make the link between the coding of the form and the coding of the model to interact to each other. Moreover, the visual basic tools such as '*userform'* and '*inputbox*' allow human to interact with the model by put in the input data into form and a model which is the result will be created. By applying this knowledge, automation process in modelling a design can be done and can be fully utilized in the industrial world.

Finally, looking for trend today most customer now demand a special editor tools in terms to purchase a product they desired. Thus this project or program can be used as the platform for that purpose. This program can produce a more systematic ordering techniques for the customer and satisfied them by showing the detail and the result of illustration of their product. After the customer has decided the design by using this program, then the part can be produced through the application such as rapid prototype or additive manufacturing.

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